## Hi,

I'm using the multiphase flow with laminar flow and phase set models for simulating the process of filing a mould and then wiping the excess of fluid with a moving wipe (or blade), as shown in the picture.

I used the moving mesh interface and added the automatic remeshing to the solver.

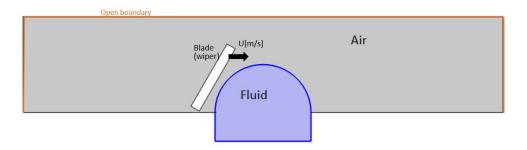
Everything works until the time in which the blade "touches" the boundary of the semi-circle I used to define the initial shape of the fluid.

If you could give me some help on how to define the simulation so it does not crash, it would be great.

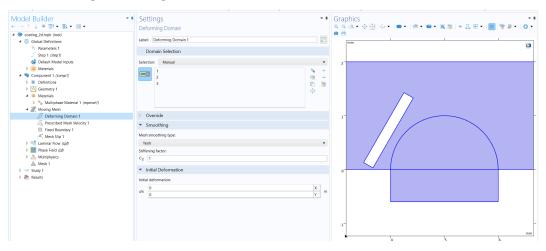
Thank you.

## More details:

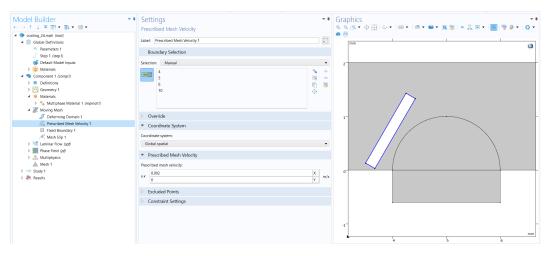
Definition of the system:



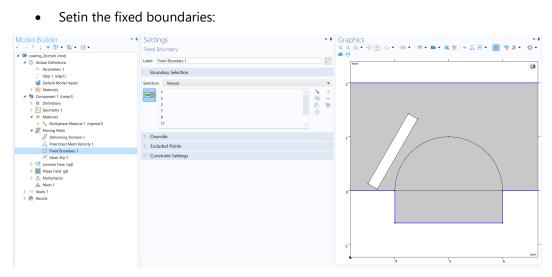
Setting the moving mesh interface:



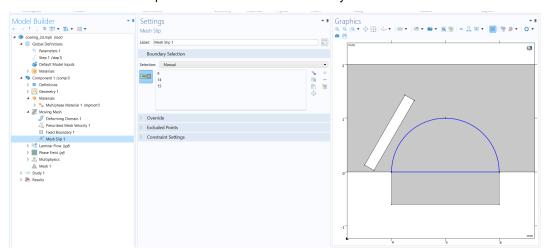
• Seting the velocity of the blade:



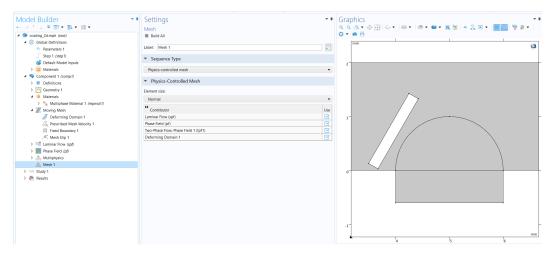
Setin the fixed boundaries:



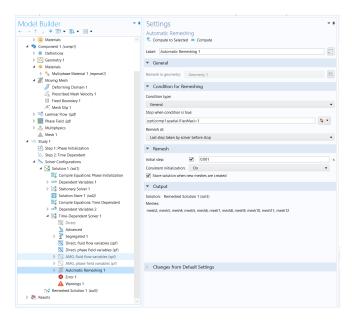
The next step I am not sure if is correct, but it was the only way it worked. It was to set the semicircle as Mesh slip or as Prescribed Mesh Velocitym with v=0.



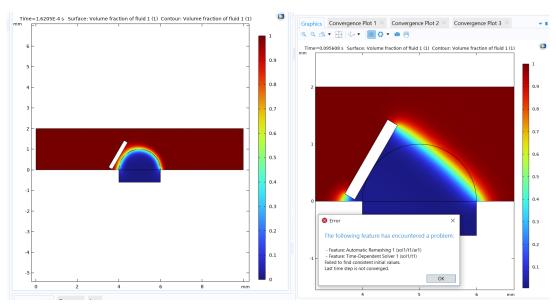
The Mesh was left in automatic.



• It was added the Automatic remeshing in the solver.



• The simulation works until the blade touches the semi-circle (which is not a physical body, it was just to define the initial fluid condition)



• This is the last mesh generated, we can see the remeshing is actuly considering the semi-circles as a bounday, and refining the mesh in the intersection. After that the simulation does not converge.

